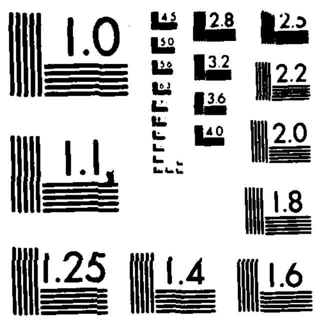
AD-A142 042    PURDUE PLANE STRUCTURES ANALYZER II A COMPUTERIZED WOOD    1/1
ENGINEERING SYSTEM 1983 VERSION(U) FOREST PRODUCTS LAB
MADISON WI   S K SUDDARTH ET AL. MAY 84 FSGTR-FPL-40

UNCLASSIFIED    F/G 9/2    NL

END
DATE
FILMED
7-84
DTIC




MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS 1963-A

## Preface

This manuscript was written as a guide to the use of the updated version of the Purdue Plane Structures Analyzer (PPSA II) computer program. For convenience, it has been divided into five sections. The first section, for those unfamiliar with PPSA, is a basic introduction to its background, use, and versatility. The second section provides instruction for the preparation of required data input. The third section presents a sample output and the fourth discusses the significance of the interaction analysis. The final section, entitled Special Topics, provides guidelines for modeling more complex structural systems.

For those interested in copying, updating, or modifying the program, Appendices A and B provide information regarding array dimensions and input format.

Arrangements can be made to obtain a copy of PPSA II by contacting

U.S.D.A. Forest Service
State and Private Forestry
c/o Forest Products Laboratory
P.O. Box 5130
Madison, Wis. 53705
(608)264-5735

## Table of Contents





**Purdue Plane Structures Analyzer II**
**1983 Version**

This program is intended to facilitate the analysis of wood structures. It gives an accurate report of structural response of the input analog. Accuracy of the analog and interpretation of structural adequacy are the responsibility of the user.

# Purdue Plane Structures Analyzer II:

## A Computerized Wood Engineering System[1]

Stanley K. Suddarth, Professor of Wood Engineering, School of Forestry, Purdue University
and
Ronald W. Wolfe, Research Engineer
Forest Products Laboratory

### Introduction

The Purdue Plane Structures Analyzer (PPSA) is a computer program for the analysis of wood structures. It is a tool for accurate prediction of the behavior of any stable wood truss or frame that can be represented by a two-dimensional analog. Introduced by Suddarth in 1972, it has found repeated application in areas of design and research. Its most frequest use has been the analysis of wood trusses.

Over the 10 years from 1972 to 1982, a number of design changes have been implemented in the National Design Specification (NDS) for Wood Construction (NFPA 1982). Design changes for member stability, combined compression-bending loads, and buckling stiffness of sheathed compression members necessitated modification of PPSA, giving rise to the development of PPSA II. This program incorporates these NDS changes as well as providing added versatility for more effective analog modeling.

Input and output formats for PPSA II are basically the same as those used in the original version (Suddarth 1972). They have undergone minor changes to provide added versatility in modeling actual structures.

The program has been set up to use a single unit of force--the pound--and a single unit of length--the inch. Any other units may be used if the same basic rule is followed in the input data. The heading labels in the printed output indicate pounds and inches, however, and must be changed to reflect any change in unit.

The question of memory storage requirements frequently governs the usefulness of a computer program. PPSA II allows for changes to be made to accommodate computer memory or problem size limitations. As the program is presently set up, it and data storage require 27 K words of memory on a UNIVAC 1110 computer.

### Program Objectives

The PPSA II program performs an analysis of an analog of a real structure. It must be recognized that computers do not substitute for engineers; computers are only a tool. It is the engineer's responsibility to recognize the limitations of the output to meet all the needs of a total design and to judge the adequacy of the analog in modeling real performance.

Structural design requires awareness of various analog assumptions. These assumptions include both details of the analog configuration and the loading arrangement. Analog configuration assumptions that may affect structural response include reactions and end fixity as well as elastic properties of individual members. Load assumptions include magnitude, location, and direction of applied loads. To vary the analog configuration, the entire set of input data must be reread. Load variations, however, require only that the new loads, along with indices that control the stress adjustment factor and interaction analysis option, be included as additional cases.

---

[1]Prepared in cooperation with Purdue University; the paper is Journal Paper 9632 of the Purdue Agricultural Experiment Station.

PPSA II can analyze joints as either rigid or pinned connections between individual members. Experience with a given type of structure may be sufficient for choosing between the simple fixity types. In doubtful cases more information can be obtained from a series of computer runs using alternate choices of rigidity. This will reveal the sensitivity of stresses and displacements in the structure to the fixity assumptions.

As a further refinement, the user can approximate partial fixity through the insertion of short fictitious members. These are rigidly connected to nodes at both ends but are made more flexible than the members joined so as to simulate axial and rotational displacements in the connection. The use of fictitious members requires knowledge of the load-displacement characteristics of the structural connection being modeled. Again, several computer runs with varied stiffness parameters can provide useful information on the sensitivity of stresses and deformation to these variations.

## Input Requirements

The matrix analysis methods used in this program require the engineer to form an analog model to simulate a real-structures response to various load conditions. A model such as that shown in figure 1 consists of a structural analog and representative loads.

### *Structural Analog*

The structural analog is read in as four data groups: member properties, node coordinates, structural assembly, and reactions. Each group requires some engineering judgment regarding either expected structural performance or program limitations.

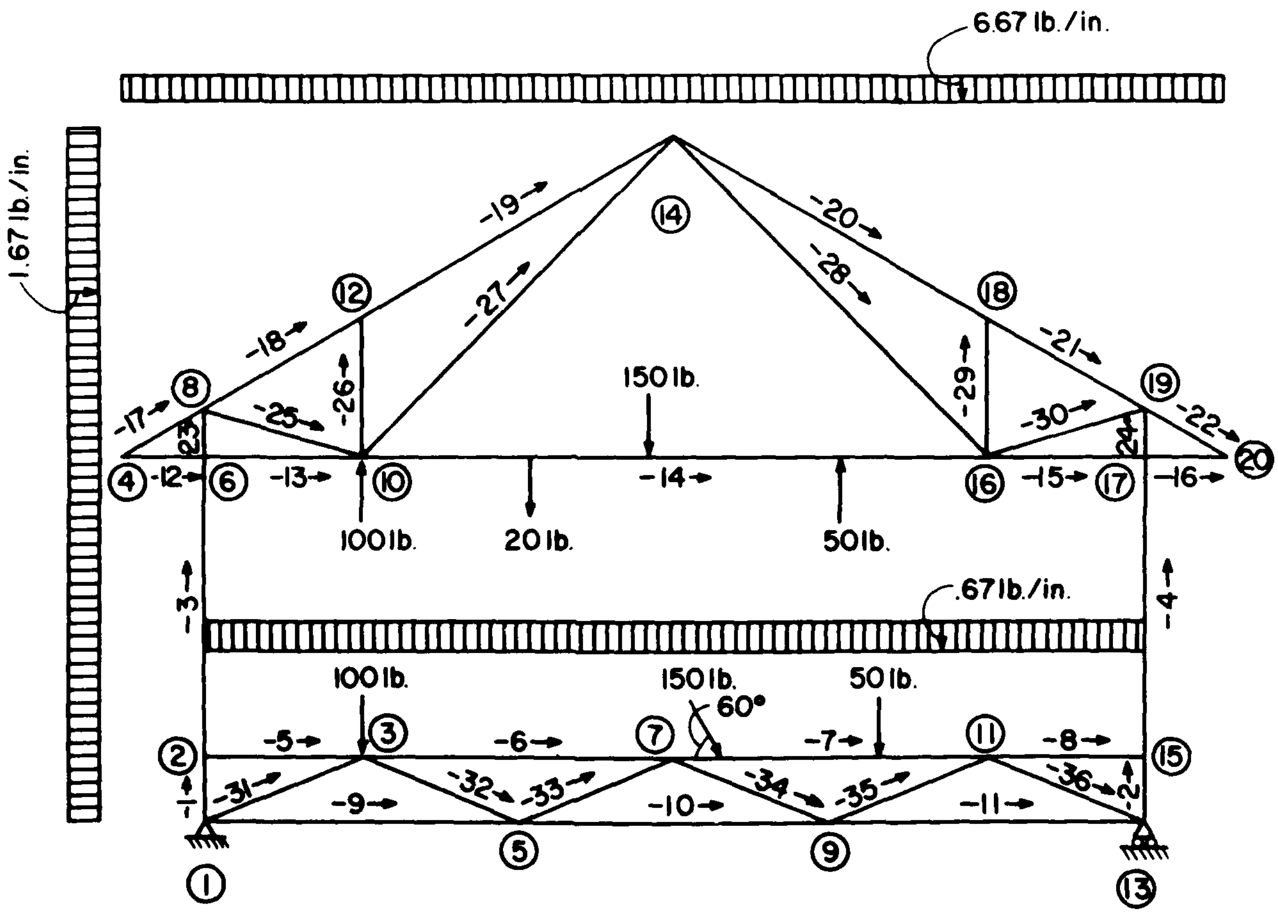



*Figure 1.--Graphic representation of an input analog for a truss frame structure. Circled numbers represent nodes. Numbers with arrows through them represent members and the arrow direction corresponds to member direction. (ML83 5587)*

Analysis of members for design interpretation is dependent upon their use, grading, and mechanical properties. The engineer has a choice of several member categories to account for lateral support conditions, effects of grading method, seasoning, and function in the structure. These categories are often of great importance in the automated application of NDS requirements within the program. If the analog member represents a real member in the structure, the user must specify dimensions, modulus of elasticity (MOE), and allowable stress in bending, compression, and tension.

The fictitious member is one that does not correspond to a real member in the structure but is used to provide versatility in modeling semirigid connections, flexible reactions, or geometric features such as joint eccentricity. Three example applications are shown in figure 2. Three sample cases using fictitious members in relating analog models to structural test data have been reported (Suddarth and Percival 1972).

Nodes, defined as the intersection of two or more members, are read into the computer as a matrix of rectangular coordinates. In figure 1, the nodes are labeled by circled numbers. Members are shown as straight lines connecting two nodes and are labeled by numbers with arrows drawn through them. The arrow indicates member direction, negative end to positive end. Assignment of member direction is primarily a convenience feature that facilitates understanding and interpretation of the output. The sign convention that has been found to work well defines positive member direction as being up or to the right. Thus member data are input by designating their negative and positive end nodes. Member input must include the member's attachment to its end nodes. Fixed or rigid node connections are conceptualized as shown in figure 3. In this instance, the node is pictured as a disk to which members A and B are rigidly connected and member C is pinned. This simple representation of a joint would transfer an in-plane bending moment from member A to member B with no effect on member C. It will transfer axial and shear loads to all connected members.

Permissible reactions (shown in fig. 4) include four general categories: PIN, ROLL, FIX, and a special type called the FIRL reaction. The purposes of the first three categories are implied by their names as traditionally used in structural analysis: A PIN reaction resists horizontal and vertical translation but offers no resistance to rotation; a ROLL reaction permits rotation and translation in one specified direction; the FIX reaction resists all movement. The FIRL reaction is identical to the ROLL except for the further restriction that it does not permit node rotation. FIRL reactions have special uses and are discussed more fully, with an example, in the Special Topics section.

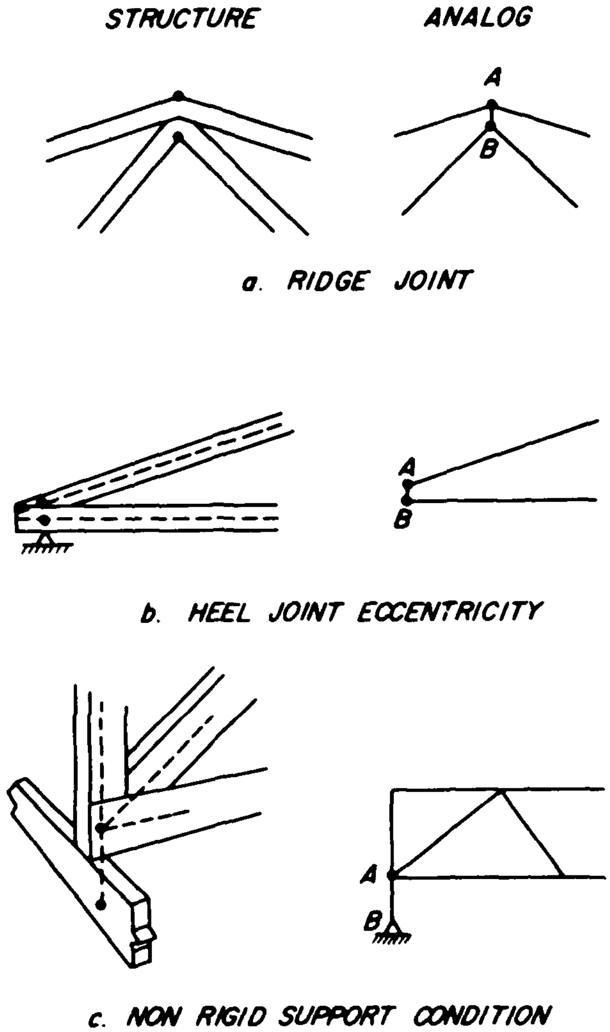



*Figure 2.--Applications of fictitious members. Applications a and b can model joint eccentricity as well as flexibility. Variation of dimensions and elastic properties enable a fictitious member to model elastic support conditions as shown in c. (ML83 5588)*

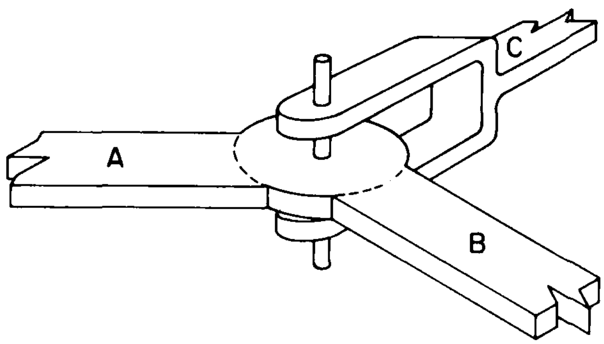



*Figure 3.--Joint mechanism representing a node as a flat disk with members A and B rigidly connected and member C pin connected. (ML83 5589)*

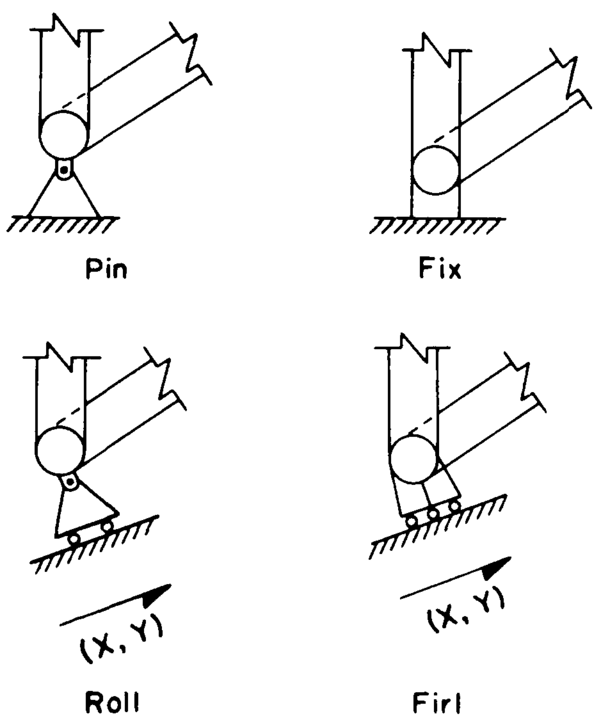



*Figure 4.--Reaction options may be selected to resist all or no rotation and all or partial translation. PIN permits rotation without translation. FIX permits no movement. ROLL permits rotation and translation in one direction. FIRL permits translation in one direction without rotation. (M139 733)*

### *Loads*

Applied loads must be classified in one of three general categories: concentrated, nodal, or uniform. Concentrated loads are point forces applied to a member; nodal loads are point forces or moments applied at a node; and uniform loads are applied continuously over the length of an individual member.

The program interprets this input in terms of system coordinate axes. The loads are read in as components oriented parallel to the horizontal and/or vertical axes of the structure, commensurate with a standard Cartesian coordinate system: positive up and to the right. For purposes of member stress analysis, the program converts these loads from a structure orientation to a member orientation, giving load components parallel and perpendicular to the loaded member length.

In the case of uniform loads, only inclined members can accept both horizontal and vertical components. Horizontal members will not accept a horizontal uniform load and vertical members will not accept a vertical uniform load.

## Program Operation

Input data are used to develop a stiffness matrix for each member, which, in turn, is incorporated into a system stiffness matrix. Derivation of the elements oı this stiffness matrix assumes a shear modulus (G) equal to 1/16 of the MOE. The value of G is then adjusted by a shear coefficient (called ALPHA in the program) of 6/5 (Orosz 1970) for rectangular sections. Both G and ALPHA have been programmed to facilitate revision if the program user is analyzing other than rectangular sections.

The matrix analysis of structures is a convenient procedure that takes advantage of the capabilities of digital computers to perform structural engineering tasks that were formerly considered economically prohibitive. Specifically, this program involves a linear stiffness method solution based on a virtual energy formulation. This topic is well established in textbook form (Przemieniecki 1968; Zienkiewicz 1967) and engineering journal articles. Because of the prolific supply of background material, this report does not discuss theory or computer program details but rather presents the instructions for using this specific matrix analysis program tailored to the needs of wood engineering.

In addition to echoing input, the standard program output includes reaction values, forces and moments at the ends of each member, the critical axial and bending stresses, maximum shear stresses, member deflections, and node displacements. Input options permit the user to eliminate unwanted items of output.

## Preparation of Program Input

User-computer communications may make use of either a keyboard computer terminal or cards for entering the analog data. In the following discussions, the term input line or line refers to the data recorded on one card or one line of type. The PPSA II program is formatted to read input data one line at a time and to identify individual values by line sequence and location in the input line. Appendix A shows input data coding forms that may be used to facilitate the input process.

The user will note that this input is quite detailed, requiring considerable preparation time. It has been kept in this form to afford maximum flexibility in a wide range of applications. Some users may wish to write their own preliminary program that takes simplified input instructions and develops this data file, utilizing their own rules for creating the analog of a specific class of structures.

Integer values are often used to number or index an item of input. Their values are expressed as whole numbers and should always be right-justified in their entry fields. A decimal point placed in an integer field would not be recognized and would be interpreted as an error. Real values, on the other hand, consist of both whole number and decimal portions. Input data formats for this program are written so that if no decimal is supplied by the user for a real variable, it is automatically supplied by the program to the right of the last character in the specified data field. Real number inputs must recognize these rules. For instance, if the real number 25 is to be entered in columns 13-17 of an input data line, it may be read in as *25.0b* or *b25.b* or *bbb25.* However, if it is input as *b25bb*, the value read will be 2500. Real values may also be read using an exponential format. In this instance, the value is separated into a mantissa and an exponent by the letter E. For example, 1 million would be read into a 7-character field as *+1.0E06.* Any numerical variables not labeled integer should be assumed to be real.

Input data have been divided into eight categories: Identification, problem size, member properties, stress factor, point coordinates, structural assembly, reactions, and loads. The following discussion describes each of these categories and makes reference to the analog shown in figure 5 to illustrate the manner in which the categories are to be entered for program execution.

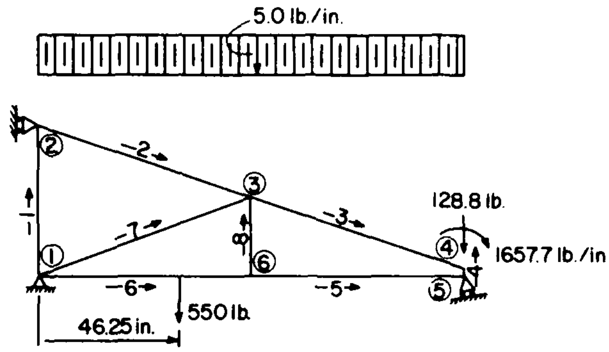



*Figure 5.--Analog diagram for a single pitched truss supported at three points and subjected to a uniform load, a concentrated load, a nodal load, and an end moment. The moment (1657.7 lb/in.) and nodal load (128.8 lb) at node 4 represent the effects of a 2-foot overhang extension of the top chord. (M139 730)*

## Identification

Any keyboard characters can be used to identify the particular analysis being processed except the sequence ENDDCALC in columns 1-8. The identification character string may occupy up to 68 columns on one line.

Example entry:

| | | | | | | | | | | | | | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 | 18 | 19 | 20 | 21 | 22 | 23 | 24 |
| E | K | A | M | P | L | E | | S | H | E | D | | T | R | A | M | E | , | | 4 | / | 1 | 2 |
| 25 | 26 | 27 | 28 | 29 | 30 | 31 | 32 | 33 | 34 | 35 | 36 | 37 | 38 | 39 | 40 | 41 | 42 | 43 | 44 | 45 | 46 | 47 | 48 |
| S | L | O | P | E | , | | 2 | X | 4 | | C | O | N | S | T | | | | | | | | |
| 49 | 50 | 51 | 52 | 53 | 54 | 55 | 56 | 57 | 58 | 59 | 60 | 61 | 62 | 63 | 64 | 65 | 66 | 67 | 68 | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | |

The sequence of characters read on this line is printed out as a heading for the program output.

## Problem Size

The Problem Size line tells the computer the size of the structure being analyzed, the amount of output wanted, and the number of load situations to be considered.

Line format:

Columns 1-2--Integer: Number of points (NP) used in the analog.
Columns 3-4--Integer: Number of members (NM) in the analog.
Columns 5-6--Integer: Number of Roll (NR) reactions used in the analog.
Columns 7-8--Integer: Number of Pin (NPIN) reactions in the analog.
Columns 9-10--Integer: Number of FIRL (NFIR) reactions used in the analog.
Columns 11-12--Integer: Number of FIX (NFX) reactions used in the analog.
Column 13
- 0 or blank = List all input (tables I-VI).
- 1 = Do not list input.

Column 14
- 0 or blank = List all output (tables VII-XII).
- 1 = List only tables VII and VIII--reactions and member end actions.
- 2 = List only table IX--analysis.
- 3 = List only tables IX and X--interaction and shear stresses.
- 4 = List only tables XI and XII--deformations and displacements.
- 5 = List only tables IX-XII.

Columns 15-16--Integer: Number of load cases (NLOAD). If only one load case is to be considered, this may be left blank. The purpose of this input is to permit repeated analyses of a structural configuration under different loads without the added time and expense of reentering member and joint information and recalculating the structural stiffness matrix. Subsequent load cases operate at higher speed at a fraction of the computing time cost of the initial load case.
Columns 17-18--Integer: Number of locations checked for stress and deformation along each member. If left blank, 24 points will be checked along each member.

Example entry:

| NP | | NM | | NR | | NPIN | | NFIR | | NFX | | INPRIN | JPRINT | NLOAD | | NDIV | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 | 18 |
| | 6 | | 8 | | 2 | | 1 | | | | | | | | | | |

## Member Properties

Member Property lines describe the member mechanical properties, sizes, and type of use.

Line format:

Columns 1-2--Integer: Member group number beginning with b1. Each variation in member size, strength, or type of use creates a so-called member group and thus requires a separate input line.
Columns 3-7--Thickness, in inches, of the member in the direction perpendicular to the plane of the structure. Columns 8-12--Depth, in inches, in the plane of the structure.
Columns 13-18--MOE, in pounds per square inch, of the member material. Because of the magnitude of this number, all MOE values are read in using an exponential format. An MOE of 1,760,000 could be written $1.76 \times 10^6$: The computer recognizes this number written as 1.76E6 in the six-column field.
Columns 19-22--Normal duration allowable bending stress in pounds per square inch (ASIGM). This and the following two entries come from lumber grade stress tables such as are found in the NDS (NFPA 1982) and are entered as whole numbers with the decimal supplied by the machine at the right-hand side of the field. When a special stress adjustment is pertinent to only one category of stress, the adjusted value should be used in the appropriate field. An example for bending stress is the 15 percent increase for repetitive member use (table 4A of NDS). Since this factor is applicable only to bending stress ($F_b$), the adjustment should be made prior to entry in this field. The program estimates this adjustment factor on the basis of the effective bending length override entry discussed later under Structural Assembly input.

The ASIGM field is left blank if the member is fictitious and not subject to shear deflection. If shear deflection is to be included in a fictitious member, any nonzero entry will suffice, although it is convenient to use a number like 9,000, which is obviously not a wood stress value.

Columns 23-26--Normal duration allowable compression stress, in pounds per square inch. This field is left blank if the member is fictitious.
Columns 27-30--Normal duration allowable tensile stress, in pounds per square inch. This field is left blank if the member is fictitious.

Column 31--Integer: Type use classification for nonfictitious members with the following code. For this discussion, Member refers to main structural members such as truss chords, and Interior Members are secondary structural members such as truss webs. This column is left blank for a fictitious member.

0 or b = Member, laterally supported as by sheathing, etc. (NDS 3.3.2) (S).
1 = Member, not laterally supported (U).
2 = Interior Member, laterally supported as by adequate bracing (SI).
3 = Interior Member, not laterally supported (UI).
4 = Truss Chord, seasoned (NDS 3.10.5) (TD).
5 = Truss Cho·d, green (NDS 3.10.5) (TG).
6 = MSR Truss Chord, seasoned (NDS 3.10.5) (TDM).
7 = MSR Truss Chord, green (NDS 3.10.5) (TGM).
8 = MSR Member, laterally supported (SM).
9 = MSR Member, not laterally supported (UM).
A = MSR Interior Member, laterally supported (SIM).
B = MSR Interior Member, not laterally supported (UIM).

Column 32--Last line indicator, which is 1 to indicate that the next line begins a new type of data and is blank otherwise.

Entries 4 through A listed for column 31 were added to the PPSA II program in response to new categorization of compression members originating with the previous version of the NDS. Special consideration is given in paragraph 3.10.5 of NDS to 2 x 4 trusses spaced 2 feet on center or less and suitably sheathed with plywood as in residential construction. Also, reduced variation in MOE for MSR lumber leads to different column formulas, as shown in Appendices G and O of the NDS.

When the member is classified as laterally supported and is detected by the computer as a compression or bending-compression member, the interaction analysis is performed with column calculations referred only to member depth in the plane of the structure. When the member is classified as not laterally supported, analyses are performed in the plane perpendicular to that of the truss as well. The most critical interaction value is reported.

Example entry:

| Member Group No (1-2) | Member Thickness (3-7) | Member Depth (8-12) | Modulus of Elasticity (13-18) | Allowable Bending Stress (19-22) | Allowable Compression Stress (23-26) | Allowable Tensile Stress (27-30) | MTYPE 1 (31) | NOM (32) |
|---|---|---|---|---|---|---|---|---|
| 1 | 1.5 | 3.5 | 1.00 E 6 | 1600 | 1450 | 1300 | 0 | |
| 2 | 1.5 | 3.5 | 1.00 E 6 | 1600 | 1450 | 1300 | 1 | |
| 3 | 1.5 | 3.5 | 1.00 E 7 | 9000 | | | | |
| 4 | 1.5 | 3.5 | 1.60 E 6 | 1200 | 1000 | 900 | 3 | 1 |

## Stress Adjustment Factor

The Stress Adjustment Factor is used to adjust normal duration stress for design load in accordance with section 2.2.5 of NDS. Any other general strength alterations applicable to all elements in the entire structure, such as the use of fire-retardant-treated lumber, can also be combined into this one factor. All allowable stresses from the previous line group are multiplied by the factor prior to use within the program.

Up to three stress adjustment factor values may be read in to allow for treatment of different load cases. The first factor is entered in columns 1-5, the second in columns 6-10, and the third in columns 11-15. An index is read in later, along with load input, to specify which factor is to be used. If no index is given, the first factor is selected.

Example entry:

| FACTOR I | | | | | FACTOR II | | | | | FACTOR III | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 |
| 1 | . | 1 | 5 | | 1 | . | 2 | 5 | | 1 | . | 3 | 3 | |

## Point Coordinates

Point Coordinate lines give the Cartesian coordinates of each point in the analog. Member lengths and direction cosines are derived by the program from these values. In setting up this coordinate system, careful selection of the origin may facilitate future revision of the analog. For example, if a complex system such as that shown in figure 1 were being used, it might be advantageous to place the origin at point 2. This would permit variations in the depth of the floor truss without affecting the node coordinates for the rest of the structure.

Line format:

Columns 1-2-Integer: Node number as shown on the analog.
Columns 3-9: The *x* coordinate of the node, in inches.
Columns 10-16: The *y* coordinate of the node, in inches.
Column 17: Last card indicator, which is 1 for the last card and blank otherwise.

Another important consideration in the assignment of node coordinates is the order in which nodes are numbered. For a given structure, all numbering systems lead to the same size square symmetric stiffness matrix. Figure 6 compares the square stiffness matrix which requires $N^2/2$ locations for storage of the upper or lower triangle to a banded form requiring only N x B storage locations. For a plane structure in which member end conditions consist of axial loads, shear, and moments:

$$N = 3 \times (\text{number of nodes}) - (\text{number of reaction constraints})$$

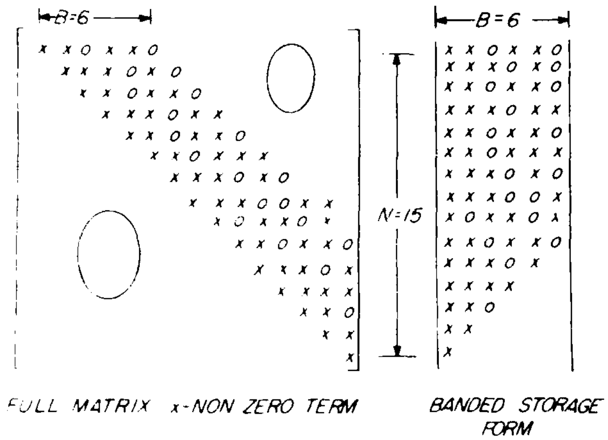



*Figure 6.--A symmetric stiffness matrix that has all nonzero terms within a distance B from the diagonal may be stored in banded form to conserve computer memory space. (ML83 5590)*

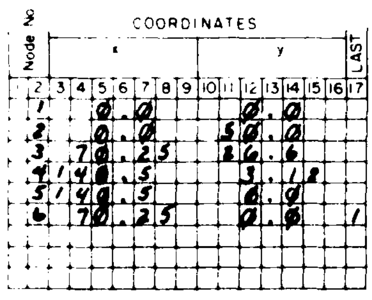
The width (B) of the banded matrix is dependent upon the member having the largest difference between end node numbers:

B = 3 × [(maximum difference) + 1]

To minimize the number of steps required for the analysis, the value of B should be kept as small as possible by numbering nodes in such a manner as to minimize the difference between member end node numbers. For example, the analog shown in figure 1 has a maximum difference of only 4. As the program is currently written, a difference greater than 11 will cause the error message:

BANDWIDTH OF STIFFNESS MATRIX IS GREATER THAN ALLOWABLE STORAGE AREA

Example entry:




## Structural Assembly

The Structural Assembly lines identify the two end nodes to which each member is connected, designate the connection as either fixed or pinned, and classify the member according to member group as listed on the member property input lines. In addition, the Structural Assembly lines give the user the option of independently specifying effective column lengths for each member in planes parallel and perpendicular to the plane of the structure and specify effective bending length for beams lacking continuous edge support and/or torsional restraint at bearing points (NDS 3.3.3 and 4.4).

Line format:

Columns 1-2--Integer: Member number.
Columns 3-4--Integer: Node location of the negative end of the member.
Columns 5-6--Integer: Node location of the positive end of the member.
Columns 7-8--Integer: Member group number to assign appropriate member properties according to the number designated in the Member Properties line.
Column 9--Condition of negative member end connection to the node:
0 = Rigidly connected.
1 = Pin connected.
Column 10--Condition of positive member end connection to the node:
0 = Rigidly connected.
1 = Pin connected.
Column 11--Last line indicator, which is 1 for the last line of member data and blank otherwise.
Columns 12-17--Effective column length, in inches, in the plane of the structure (COLH) (optional override).
Columns 18-23--Effective column length, in inches, perpendicular to the plane of the structure (COLT) (optional override).
Columns 24-29--Effective unsupported bending length, in inches (BLE) (optional).

Fixing intersecting members to a node causes them to rotate together and share moments with other members that may likewise be fixed to the same node.
Pinning a member end at a node causes a zero moment at that member end and creates independence between rotation of the node and rotation of the member end.
The original PPSA (Suddarth 1972) required that at least one member end be fixed to each node in the analog to avoid an arithmetic error within the computer. PPSA II

has been structured to allow a node to be free of fixation from any member. The node is then fixed against rotation in the analysis and a zero rotational displacement will consequently be reported.

Specified values for effective column length can be used in the analysis of column buckling. These input options override the automated effective column length determinations made by the computer. An example of a use for this option would be the analysis of truss chords laterally supported only at spaced intervals by purlins. If a member is defined as being laterally supported, no analysis is performed on column buckling in the perpendicular-to-plane direction. When lateral support is less than complete, the user may enter the effective column length in the perpendicular plane due to purlin support and declare the member laterally unsupported. This will cause an analysis to occur in both directions using the engineer's specified effective column length.

Structural cases can arise in which the end or ends of a compression member are not confined to small displacements in a direction perpendicular to the critical column axis. This requires the engineer to use the appropriate optional column length override (COLH or COLT) to input a suitable value for the effective column length.

Bending members with inadequate rotational or lateral displacement restraint and a depth-to-thickness ratio greater than 1 are subject to out-of-plane displacement (NDS 3.3.3 and 4.4). For these cases, the allowable bending stress should be adjusted by a slenderness factor (NDS 3.3.3.4). PPSA II calculates this factor, using the BLE input (cols. 24-29), for any member having a positive 'effective unsupported bending length' and a 'not laterally supported' member group designation (MTYPE = 1, 3, 9, or B on Member Properties lines).

Member numbering has no effect on storage or computation time. A numbering system should be selected on the basis of convenience of output clarity. For the analog shown in figure 1, for instance, members are numbered consecutively with groups (i.e., wall studs, truss chords, and truss web members). An alternative to this would be a floor, wall, roof division. The shed roof being used as a demonstration model (fig. 5) is simply numbered in a clockwise sequence beginning at the lower left reaction.

Example entry.

| Member No (1-2) | Neg End Node (3-4) | Pos End Node (5-6) | Member Group (7-8) | End Fixation Neg (9) | End Fixation Pos (10) | LAST (11) | Effective Column Lengths (Engineer's Override Option) In-Plane (in) (12-17) | Effective Column Lengths (Engineer's Override Option) Perp to Plane (in) (18-23) | Effective Bending Length (in) (24-28) |
|---|---|---|---|---|---|---|---|---|---|
| 1 | 1 | 2 | 4 | | | | | | |
| 2 | 2 | 3 | 1 | | | | | | |
| 3 | 3 | 4 | 1 | | | | | | |
| 4 | 5 | 4 | 3 | | | | | | |
| 5 | 6 | 5 | 2 | | | | | | |
| 6 | 1 | 6 | 2 | | | | | | |
| 7 | 1 | 3 | 4 | 1 | 1 | | | | |
| 8 | 6 | 3 | 4 | | | 1 | | | |

Structural assembly entries are for the shed frame (fig. 5). Note that no overrides on column or beam length have been specified in this example.

## Reactions

These lines locate Reactions at their respective nodes, give the types, and designate the direction of roller movement when pertinent.

Line format:

Columns 1-2--Integer: The node number at which the reaction occurs.
Columns 3-6--Name: The name of the kind of reaction from among the four types: PIN, ROLL, FIX, and FIRL.
Columns 7-11--Horizontal component of the vector describing the direction in which rollers are free to move. This field is blank in the case of PIN or FIX.
Columns 12-16--Vertical component of the vector describing the direction in which rollers are free to move. This field is blank in the case of PIN or FIX.
Column 17--Last card indicator, which is 1 for the last card and blank otherwise.

Example entry

| Reaction Node No (1-2) | Reaction Type (3-6) | Horizontal Component (7-11) | Vertical Component (12-16) | LAST (17) |
|---|---|---|---|---|
| 1 | PIN | | | |
| 2 | ROLL | Ø.Ø | Ø.Ø | |
| 5 | ROLL | 12.Ø | 2.Ø | 1 |

In this shed frame example, the roller reaction at node 2 permits rotation, moves only in the vertical direction, and resists movement due to any loads in the horizontal direction. The roller at node 5 moves on a 2 on 12 slope. The pin reaction at node 1 resists movement due to either horizontal or vertical loads but permits rotation.

## Load Information and Interaction Interpretation

For each load case the user must specify load type, method of analyzing the interaction of axial and bending stresses, and an allowable stress adjustment factor. The Load Information and Interaction Interpretation line provides these specifications through the use of option indices. The first three columns designate yes/no options for concentrated, uniform, and nodal loads, respectively. Columns 4 and 5 each permit up to four options for interaction interpretation and allowable stress modification respectively.

Line format:

Column 1--Concentrated load indicator: Enter 1 if there is at least one concentrated load on the structure; otherwise, enter 0 or leave blank.
Column 2--Uniform load indicator: Enter 1 if there is at *least one uniform load on the structure; otherwise,* enter 0 or leave blank.
Column 3--Nodal load indicator: Enter 1 if there is at least one force or moment applied directly to a node in the structure.
Column 4--Interpretation of interaction equation option (ITP): PPSA II allows for some engineering choice in this matter with the optional entries 0, 1, 2, or 3. Each real member in the structure is treated the same way according to this option. The choice relates primarily to where the stresses for the interaction formula, NDS 3.10, are calculated. The consequences of choice appear in table IX of the output and are discussed in more detail in a later section.

Option 0.--This is the most conservative option and calculates the interaction value using bending stress at the point of maximum moment and axial stress at the point of maximum compression. The two maxima do not have to occur at the same point.

Option 1.--The interaction equation is calculated using the maximum bending stress and the axial stress at the same location as the maximum bending stress. This option is set up so that a constant or zero moment stress over the full length of the member will cause the interaction value to be calculated at the point of maximum axial stress. Cases in which moment is constant over only part of the member should be carefully examined for a more critical interaction value than that reported.

Option 2.--The interaction equation is calculated using the maximum axial stress and the bending stress at the same location as the maximum axial stress.
This option is set up so that a constant or axial stress over the full length of the member will cause the interaction value to be calculated at the point of maximum bending stress. Cases in which axial force is constant over only a part of the member should be carefully examined for a more critical interaction value than that reported.

Option 3.--This option closely follows the methodology underlying the development of the Truss Plate Institute design specification for 1978. It pertains only to structures of the truss type with uniform loading. Loads may be imposed at nodes but concentrated loads directly applied to members are not allowed. The interaction equation is calculated at each member end node using the short column interpretation given in NDS 3.10.2.2. A third interaction value is calculated at the point of maximum moment within the span using the compression stress at that point and an automatically calculated effective column length. The largest of the three interaction values is used for the member.

Column 5--Stress factor index: This index corresponds to the order in which the stress factors are given on the *stress adjustment factor line.* A blank, Ø, or 1 will cause the use of the first value. A 2 or 3 will result in the use of the second or third value, respectively, and any index greater than 3 will result in a stress factor of 1.0.

Example entry:

| ICON | IUN | ISYST | ITP | SFT |
|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 |
| 1 | 1 | 1 | | |

The example structure is loaded by at least one of each load type, the interaction analysis will use the maximum bending moment and maximum compression force values in each member span, and the first stress factor read in will be used.

## Concentrated Loads

Concentrated Load lines identify the member carrying the load, its magnitude and direction, and its location along the member. Each member may carry up to three concentrated loads. These are numbered sequentially beginning at the negative end of the member. Each load requires a line of input.

Line format:

Columns 1-2--Integer: Member number carrying the load.
Column 3--Integer: Load number 1, 2, or 3.
Columns 4-9: Horizontal or *x* component of the load, in pounds (positive to right).
Columns 10-15: Vertical or *y* component of the load, in pounds (positive up).
Columns 16-21: Distance from the negative end of the member to the load point.
Column 22: Last card indicator, which is "1" for the last line and blank otherwise.

Example entry:

| Member No | | Load No | Horizontal Component | | | | | | Vertical Component | | | | | | Load Dist from Neg End (in) | | | | | | NEXT |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 | 18 | 19 | 20 | 21 | 22 |
| | 6 | 1 | | | Ø | . | Ø | | - | 5 | 5 | Ø | . | Ø | | 4 | 6 | . | 2 | 5 | 1 |

## Uniform Loads

Uniform Load lines identify the loaded member and give the horizontal and vertical uniform load components for that member (in pounds per unit length) where length is expressed in the same units used in defining node coordinates.

Line format:

Columns 1-2--Integer: Member number.
Columns 3-12: Horizontally applied uniform load, in pounds per inch (positive to right).
Columns 12-22: Vertically applied uniform load, in pounds per inch (positive to up).
Column 23: Last line indicator, which is 1 for the last line and blank otherwise.

Example entry:

| Member No | | Horizontal Component | | | | | | | | | | Vertical Component | | | | | | | | | | NIX |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 | 18 | 19 | 20 | 21 | 22 | 23 |
| | 2 | | Ø | . | Ø | | | | | | | - | 5 | . | 0 | | | | | | | |
| | 3 | | Ø | . | Ø | | | | | | | - | 5 | . | 0 | | | | | | | 1 |

## Nodal Loads

Nodal Load lines identify the node at which the load is applied, the type of load, horizontal or vertical force (in pounds), or a moment (in inch-pounds), and the magnitude. One line is required for each load.

Columns 1-2--Integer: Node number carrying the load concerned.
Column 3: The type of load which is coded as 1 for a force in *x* direction, 2 for a force in the *y* direction, and 3 for a moment.
Columns 4-11: The force (in pounds) or moment (in pound-inches), whichever is the case. Forces are positive to the right and upward to agree with the x,y system. Positive moments are counterclockwise.
Column 12: Last card indicator, which is "1" for the last nodal load card and blank otherwise.

Example entry:

| Node No | | Load Direction | Load (lb or in - lb) | | | | | | | | NON |
|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 |
| | 4 | 2 | - | 1 | 2 | 8 | . | 8 | | | |
| | 4 | 3 | - | 1 | 6 | 5 | 7 | . | 7 | | 1 |

If more than one load case is to be used, as indicated by the entry in NLOAD on the problem-size line, the necessary information is entered after the last card of the previous load case. Required input for subsequent load cases begins with the Load Information and Interaction Interpretation line and continues through to the end of each set of load input lines.

The last line of input data must contain ENDDCALC in columns 1-8. This entry is required for normal termination of program operation.

## Example of Program Output

Results of the computer analysis performed using the PPSA II program are presented in the form of 12 tables of output. The first six tables summarize the analog being considered and the last six provide an analysis of stresses and deformations that would result from the loads imposed on the analog.

## Analog Summary

The computer report begins by identifying the problem (ID input line) and indicating basic statistics of the analog. Table I shows properties of component materials being considered for the structure along with the classification of member use type.

Tables II, III, IV, and V provide a description of analog physical properties. These include node coordinates, member layout, and the locations and types of reactions. Table V provides a sequential listing of members along with their length, optional overrides on column lengths, and effective lengths for beam slenderness calculations. Finally, table VI lists the loads carried, along with their locations and directions.

Example tables I-VI:

```
EXAMPLE SHED ROOF,4/12 SLOPE
=====================================================================


NUMBER OF NODES                          =  6

NUMBER OF MEMBERS                        =  8

NUMBER OF ROLLER SUPPORTS                =  2

NUMBER OF PINNED SUPPORTS                =  1

NUMBER OF FIRL SUPPORTS                  =  0

NUMBER OF FIXED SUPPORTS                 =  0

NUMBER OF LOADING ARRANGEMENTS           =  1
```

TABLE I — ALLOWABLE MEMBER STRESSES IN PSI
NORMAL LOAD DURATION

| MEMBER GROUP | USE TYPE | ALLOWABLE BEND | COMP | TENS | WIDTH | DEPTH | MODULUS OF ELASTICITY |
|---|---|---|---|---|---|---|---|
| 1 | S | 1600. | 1450. | 1300. | 1.500 | 3.500 | 1.80+006 |
| 2 | U | 1600. | 1450. | 1300. | 1.500 | 3.500 | 1.80+006 |
| 3 | FI | 9000. | 0. | 0. | 1.500 | 3.500 | 1.80+007 |
| 4 | UI | 1200. | 1000. | 900. | 1.500 | 3.500 | 1.60+006 |

TABLE II NODE COORDINATES

| NODE NO. | X-COORD (IN) | Y-COORD (IN) |
|---|---|---|
| 1 | .000 | .000 |
| 2 | .000 | 50.010 |
| 3 | 70.250 | 26.600 |
| 4 | 140.500 | 3.180 |
| 5 | 140.500 | .000 |
| 6 | 70.250 | .000 |

TABLE III MEMBER LAYOUT

| MEMBER | NEGATIVE END | | POSITIVE END | | MEMBER |
|---|---|---|---|---|---|
| NUMBER | NODE | CONDITION | NODE | CONDITION | GROUP |
| 1 | 1 | RIGD | 2 | RIGD | 4 |
| 2 | 2 | RIGD | 3 | RIGD | 1 |
| 3 | 3 | RIGD | 4 | RIGD | 1 |
| 4 | 5 | RIGD | 4 | RIGD | 3 |
| 5 | 6 | RIGD | 5 | RIGD | 2 |
| 6 | 1 | RIGD | 6 | RIGD | 2 |
| 7 | 1 | PNND | 3 | PNND | 4 |
| 8 | 6 | RIGD | 3 | RIGD | 4 |

TABLE IV REACTION CONDITIONS

| NODE NUMBER | REACTION TYPE | HORIZ DISPL | VERT DISPL |
|---|---|---|---|
| 1 | PIN | .00 | .00 |
| 2 | ROLL | .00 | 1.00 |
| 5 | ROLL | 12.00 | 2.00 |

TABLE V   MEMBER LENGTH PROPERTIES

| MEMBER NUMBER | ANALOG LENGTH (IN) | IN PLANE COLUMN (IN) | PRP PLANE COLUMN (IN) | BEAM LENGTH (IN) |
|---|---|---|---|---|
| 1 | 50.010 | -NA- | -NA- | -NA- |
| 2 | 74.048 | -NA- | -NA- | -NA- |
| 3 | 74.051 | -NA- | -NA- | -NA- |
| 4 | 3.180 | -NA- | -NA- | -NA- |
| 5 | 70.250 | -NA- | -NA- | -NA- |
| 6 | 70.250 | -NA- | -NA- | -NA- |
| 7 | 75.117 | -NA- | -NA- | -NA- |
| 8 | 26.600 | -NA- | -NA- | -NA- |

LOADING 1

THE STRESS ADJUSTMENT FACTOR IS 1.15

TABLE VI A
THE STRUCTURE HAS CONCENTRATED LOADS AS FOLLOWS

| MEMBER NUMBER | LOAD NUMBER | HORIZ COMP (LBS) | VERT COMP (LBS) | LOAD DIST FROM NEG END (IN) |
|---|---|---|---|---|
| 6 | 1 | .0 | -550.0 | 46.25 |

TABLE VI B
THE STRUCTURE HAS UNIFORM LOADS AS FOLLOWS

| MEMBER NUMBER | HORIZ COMP (PLI) | VERT COMP (PLI) |
|---|---|---|
| 2 | .000 | -5.000 |
| 3 | .000 | -5.000 |

TABLE VI C
THE STRUCTURE HAS NODE LOADS AS FOLLOWS

| NODE NUMBER | DIRECTION OF LOAD | LOAD IN LBS OR IN-LBS AS APPROPRIATE |
|---|---|---|
| 4 | 2 | -128.8 |
| 4 | 3 | -1657.7 |

## Analysis Summary

The results follow the data in three sections; the first section (table VII) gives reactions; the second (tables VIII, IX, and X) summarizes strength analysis; and the third (tables XI and XII) provides deflection information.

Table VII lists the reaction locations, magnitude, and direction (horizontal, vertical, or moment). It also checks to see if the sum of forces and moments at the reactions are equal in magnitude and opposite in direction to the applied loads. The reactions are determined by summing end actions for all member ends intersecting each reaction node and thus would contain any errors in the matrix analysis of the total structure. An excellent solution check is provided by summing the resulting horizontal and vertical reaction forces as well as their moments about the x-y origin and comparing these values with corresponding sums from input loads. These sums and the difference between corresponding load and reaction values are printed out immediately following table VII.

Example table VII:

```
******************************** RESULTS ********************************

TABLE VII                                  REACTIONS
-------------------------------------------------------------------------

          REACTION      HOR. COMP.      VERT. COMP.       MOMENT
           NODE           (LBS)            (LBS)         (IN-LBS)

             1           896.178         1005.070             .000
             2          -833.472             .000             .000
             5           -62.705          376.230             .000
          -------------------------------------------------------

          SUM OF LOADS        .0          -1381.3         -9.5+004
          SUM OF REACTS       .0           1381.3          9.5+004
          DIFFERENCE          .0               .0               .0
-------------------------------------------------------------------------
```

Member strength analysis begins with table VIII. This table gives the axial and shear forces as well as bending moments occurring at the ends of each member. Table values are most conveniently utilized if they are referred to a standard member position that involves a horizontal orientation with the negative end of the member located to the left. The sign convention used for shear and axial forces is positive upward and to the right on a member in standard position. Therefore, axial load signs will be the same as the member end for tensile members and opposite to member end for compression members--i.e., negative on the negative end for tension member, etc. Bending moment signs follow the common mathematical right-hand rule: counterclockwise is positive. Figure 7 provides a graphic representation of the member end action analysis of member 3 as interpreted from the member 3 line in table VIII.

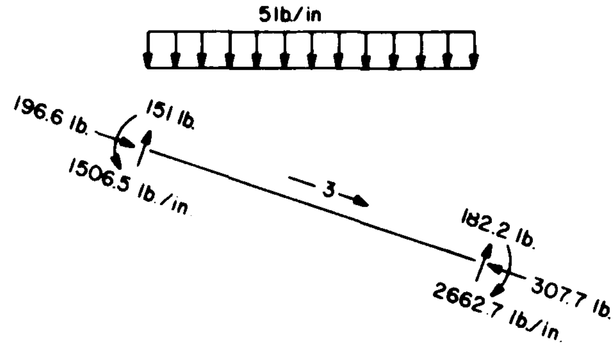



*Figure 7.--Member 3 from the analog of the example truss is shown with its load and end actions. Directions and magnitudes of the end actions are obtained from output table VIII. (ML83 5591)*

Example table VIII:

*** STRENGTH ANALYSIS ***

TABLE VIII MEMBER END ACTIONS

| MEMBER NUMBER | LOCATION | AXIAL (LBS) | SHEAR (LBS) | MOMENT (IN-LBS) |
|---|---|---|---|---|
| 1 | NEG END | 418.043 | -81.173 | -2401.636 |
| | POS END | -418.043 | 81.173 | -1657.831 |
| 2 | NEG END | -845.877 | 158.765 | 1657.831 |
| | POS END | 734.830 | 174.470 | -2239.288 |
| 3 | NEG END | 196.611 | 150.992 | 1506.153 |
| | POS END | -307.700 | 182.228 | -2662.691 |
| 4 | NEG END | 398.990 | 234.272 | -260.005 |
| | POS END | -398.990 | -234.272 | 1004.991 |
| 5 | NEG END | -171.567 | 22.761 | 1338.925 |
| | POS END | 171.567 | -22.761 | 260.005 |
| 6 | NEG END | -316.166 | 158.712 | 2401.636 |
| | POS END | 316.166 | 391.288 | -4452.126 |
| 7 | NEG END | 1209.546 | .000 | .000 |
| | POS END | -1209.546 | .000 | .000 |
| 8 | NEG END | -414.049 | 144.599 | 3113.201 |
| | POS END | 414.049 | -144.599 | 733.136 |

Table IX of the output gives the results of the interaction analysis. This analysis is dependent upon the member type, the loads it carries, its length-to-depth ratios in or normal to the plane of the structure, and the interaction analysis option (ITP) chosen on the Load Information and Interaction Interpretation input line.

The first three columns provide preliminary information. Column 1 identifies the member by its number and group classification. Column 2 provides space for a footnote reference if additional information is needed to evaluate member design. The third column, labeled MAX INT VAL, gives the maximum value of the interaction equation (combined stress index) within the member if both axial and bending stresses are present. If only axial or bending stress is present, the interaction column contains the ratio of actual stress to allowable stress. The interaction concept is treated in NDS 3.10 and is discussed in more detail in the following section.

To find the maximum interaction value and the maximum shear stresses, bending, axial, and shear stresses are calculated at equally spaced locations along the length of each nonfictitious member. The number of locations analyzed is determined by the value read into columns 15-16 of the Problem Size input line. *The input default in this case results in analysis at* 24 locations. Maximum values along with their locations are then printed out in tables IX and X of the output report. If a maximum stress value occurs more than once, the location reported is usually that nearest the positive end of the member.

The designer will note that many useful facts can be derived from table IX. For instance, the member 6 portion of the lower chord (fig. 2) is more heavily loaded, having an interaction value of 0.88 (table IX). In the upper chord, on the other hand, members 2 and 3 have relatively low interaction values--0.491 and 0.509--suggesting that a lower quality lumber may be used at these locations.

Footnotes such as that referenced for member 7 also provide additional information. In the case of member 7, the footnote tells the user that the critical stress case occurs perpendicular to the plane of the structure.

Stresses reported in the table are the values required for the interaction option selected. They have algebraic signs following the convention that axial tension is positive and bending is positive when the tensile stress is along the bottom edge of a member placed in standard position (horizontal with the negative end to the left).

Table IXA has been included to give supplementary information including analog length and the final adjusted allowable stresses used in the calculation of the combined stress index reported in table IX.

Table X reports maximum shear stress in each member from analyses made at the same points where bending and axial stresses have been determined. If the same maximum is repeated, the location reported is that nearest the positive end of the member.

Example table IX:

TABLE IX INTERACTION ANALYSIS

USING MAX MOMENT AND MAX FORCE

| MEMBER NUMBER - TYPE | ADDNL INFO | MAX INT VAL | LOC FROM NEG END MAX M (IN) | LOC FROM NEG END MAX P (IN) | MAXIMUM STRESS BENDING (PSI) | MAXIMUM STRESS AXIAL (PSI) | L/D |
|---|---|---|---|---|---|---|---|
| 1-UI | *** | .686 | -NA- | 50.01 | -NA- | -79.63 | 26.672 |
| 2-S | | .505 | 74.05 | .00 | -731.20 | 161.12 | 21.157 |
| 3-S | | .509 | 74.05 | 74.05 | -869.45 | -58.61 | 11.400 |
| 5-U | | .259 | .00 | 70.25 | -437.20 | 32.68 | 46.833 |
| 6-U | | .880 | 44.96 | 70.25 | 1545.81 | 60.22 | 46.833 |
| 7-UI | *** | .770 | -NA- | 75.12 | -NA- | -230.39 | 40.063 |
| 8-UI | | .813 | .00 | 26.60 | -1016.56 | 78.87 | 14.187 |

***INTERACTION VALUE CRITICAL FOR PERPENDICULAR PLANE

TABLE IXA MEMBER ANALYSIS DATA

FINAL ADJUSTED STRESSES

| MEMBER NUMBER | LENGTH (IN) | FAXIAL (PSI) | FBEND (PSI) | FBEND PRIME (PSI) |
|---|---|---|---|---|
| 1 | 50.010 | -674.73 | 1380.00 | 1380.00 |
| 2 | 74.048 | 674.73 | 1840.00 | 1840.00 |
| 3 | 74.051 | -1627.64 | 1840.00 | 1840.00 |
| 5 | 70.250 | 1627.64 | 1840.00 | 1840.00 |
| 6 | 70.250 | 1627.64 | 1840.00 | 1840.00 |
| 7 | 75.117 | -299.06 | 1380.00 | 1380.00 |
| 8 | 26.600 | 299.06 | 1380.00 | 1380.00 |

Example table X:

TABLE X SHEAR STRESS ANALYSIS

| MEMBER | MAX. SHEAR STRESS (PSI) | LOC. FROM NEG. END (IN) | MEMBER LENGTH (IN) |
|---|---|---|---|
| 1 | -23.19 | 50.010 | 50.010 |
| 2 | -49.85 | 74.048 | 74.048 |
| 3 | -52.07 | 74.051 | 74.051 |
| 5 | 6.50 | 70.250 | 70.250 |
| 6 | -111.80 | 70.250 | 70.250 |
| 7 | .00 | 75.117 | 75.117 |
| 8 | 41.31 | 26.600 | 26.600 |

The last two tables of the computer report deal with deformation. Table XI gives maximum member deflections and table XII lists horizontal, vertical, and rotational displacements of nodes.

For table XI, deflections are calculated at each of the locations along the member for which interaction values and shear stresses have been calculated. The maximum value and its location are reported. Although shear modulus is recognized in the formulation of the structural stiffness matrix, it is not used in the calculation of maximum member deflections reported in table XI. The sign convention used for deflection tabulations is mathematically standard, designating an upward deflection as positive when the member is in standard position.

Node displacements are reported in table XII. Positive translational displacements are upward and to the right. The rotational displacements have been reported in exponential form to accommodate the wide ranges of values that may be encountered. For example, 1 degree would be given as 1.745E-02 radians. The sign convention for rotational displacements interprets counter-clockwise as positive. The calculation of node displacements includes the influence of shear deflection.

Example table XI:

*** DEFLECTION ANALYSIS ***

TABLE XI MAXIMUM MEMBER DEFLECTIONS

| MEMBER | MAX. DEFL. (IN) | LOC. FROM NEG. END (IN) | MEMBER LENGTH (IN) |
|---|---|---|---|
| 1 | -.020 | 14.003 | 50.010 |
| 2 | -.058 | 38.505 | 74.048 |
| 3 | -.049 | 29.620 | 74.051 |
| 5 | -.028 | .000 | 70.250 |
| 6 | -.156 | 39.340 | 70.250 |
| 7 | -.024 | 75.117 | 75.117 |
| 8 | .012 | 10.640 | 26.600 |

Example table XII:

TABLE XII NODE DISPLACEMENTS

| NODE NUMBER | DISPLACEMENT HORIZONTAL (IN) | DISPLACEMENT VERTICAL (IN) | DISPLACEMENT ROTATIONAL (RADIANS) |
|---|---|---|---|
| 1 | .000 | .000 | -.329-002 |
| 2 | .000 | -.002 | -.112-002 |
| 3 | -.002 | -.027 | -.294-003 |
| 4 | .005 | .001 | -.510-003 |
| 5 | .004 | .000 | -.530-003 |
| 6 | .002 | -.028 | .340-002 |

Data from the deflection tables can be combined to indicate how the member moves in space and deforms under load. This has been done for member 3 in figure 8. This illustration of member 3 before and after loading uses an exaggerated scale to emphasize the effects of the loads on node displacement and member deflection. If the deflection of a particular point in a member is needed, that information can be obtained directly by constructing the analog with a node at the location in question and rigidly connecting it to the adjoining segments of the member. The desired displacement will then be given in table XII, but automated equivalent column length calculations underlying values reported in table IX may be incorrect in such a case, and the stress information from this table should be interpreted accordingly.

This completes the analysis of the example shed roof truss. If the value of NLOAD in columns 13-14 on input card number 2 is set at a value greater than 1, the program will proceed to read in the next set of load data for the previously defined analog. Output consisting of tables VII through IX or a selected sheet will be produced for each load case. Once all load cases have been processed, the program will try to read another complete set of analog data. When there are no more data to be read, the line following the last load line must have the letters ENDDCALC in the first eight columns to terminate the program. This series of letters is always used by the program for normal termination of operations and must always be present as the last line in any input data file.

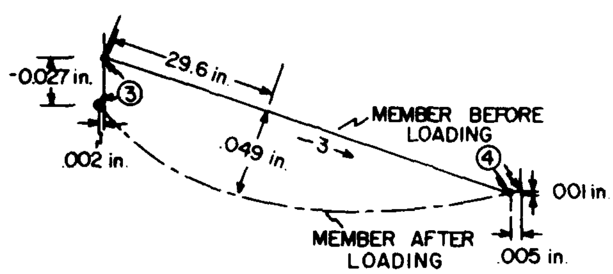



*Figure 8.--Member 3 from the analog of the example truss is shown in the before and after loading position with deflections drawn to an exaggerated scale. The node displacement and maximum member deflection (tables XI and XII) provide the data for this sketch. (ML83 5592)*

Table IX provides information regarding the maximum stress interaction value (combined stress index) for each member. The interaction analysis option (ITP), selected on the Load Information and Interaction Interpretation line, is printed out immediately below the table title. It also provides additional information, in the form of footnotes, pertaining to members that may require additional consideration beyond the scope of the PPSA II program.

Information presented in this table is useful for a wide range of applications, but its limitations must be understood. This discussion summarizes the assumptions and analysis procedures used to estimate the values presented in table IX.

## Interaction Analysis

The values reported in column 4 of table IX are calculated using one of three equations taken from NDS 3.10:

Bending-Tension combined stress index

$$I_t = \frac{f_t}{F_t} + \frac{f_b}{F_b} \qquad (1)$$

Stress difference

$$SD = \frac{f_b - f_t}{F_b'} \qquad (2)$$

Bending-Compression combined stress index

$$I_c = \frac{f_c}{F_c'} + \frac{f_b}{F_b' - Jf_c} \qquad (3)$$

where $f_b$, $f_t$, and $f_c$ are imposed bending, tensile, and compressive stresses, respectively, determined according to the ITP options included in the discussion of the Load Information and Interaction Interpretation input line, and

$F_t$ and $F_b$ are allowable tensile and bending, respectively, taken from Member Properties lines and adjusted by the value given in the Stress Factor line. $F_b$ is further limited by the size factor given in NDS 4.3.4 and 5.3.4 if its depth exceeds 12 inches.

$F_c'$ depends on column length adjustments (NDS 3.7 and 3.10) and may range from the allowable compression stress $F_c$ (adjusted by the Stress Factor) to an allowable stress for long columns.

The variable $F_b'$ is the modified bending stress accounting for a slenderness factor in members having less than complete lateral support on the compression side. NDS 3.3.3 gives information to determine the effective length, $\ell e$, which is input as BLE in columns

24-29 of the Structural Assembly lines. $F_b'$ may be equal to or less than $F_b$ depending on the value of the effective length, beam cross-sectional dimensions, and MOE. A further requirement is that $F_b'$ determined in this manner is also limited not to exceed $F_b$ adjusted by the size factor given in NDS 4.3.4 and 5.3.4. This is automated within the program utilizing the inputs of member width, depth, and BLE. An appropriate value of $F_b'$ is produced for use in equations (2) and (3). If no value is entered for BLE, it is taken as zero.

The variable J (NDS 3.10.2) is an interpolation value used to modify equation (3) for intermediate columns. Intermediate columns have $\ell/d$ ratios between 11 and k, in which k is the transition ratio between intermediate and long columns (NDS 3.10.5). Within this range, J varies as a linear function of $\ell/d$ ratio from zero for short columns ($\ell/d \leq 11$) to 1 for long columns ($\ell/d > k$) (NDS 3.7, 3.10.5 and appendix G).

## Effective Column Length

The effective column length, as defined in NDS appendix N, is the distance between two points along the length of a compression member between which it is assumed to buckle in the shape of a sine wave.

A value for effective column length is needed in the *plane of the structures for all members subjected to* compression stress. It may be supplied as COLH in the Structural Assembly input by the user. If COLH is entered as zero or blank, an automated procedure determines an effective column length limited to the analog length of a Member or 0.8 times the analog length of an Interior Member. The procedure assumes that the member ends are constrained against significant translational displacement so as to restrict the effective column length within the analog member length. Truss chord and web members usually fall into this category. The method deals with each compression member in the example fashion given for member N in figure 9. The location of N within the structure is shown in figure 9a along with its end rotational stiffnesses, $K_N$ and $K_p$, which are quantities determined within the matrix structural analysis. An equivalent beam structure on one pin and three roller supports places member N in a similar end restraint condition (fig. 9b). The values $K_N$ and $K_p$ are duplicated by suitable lengths for the end spans. An axial load, P, is placed at the roller end of N acting against the pin support at its other end. A stability analysis of the figure 9b structure yields the buckling load, P. Finally, a simple column is created (fig. 9c) of such length, L, that P is the buckling load. L is then utilized as the effective column length for in-plane analysis of member N.

A value for effective column length may also be needed perpendicular to the plane of the structure. If the member is declared laterally supported, this column length is automatically set at 11 (short category). If COLT in the Structural Assembly input for a laterally unsupported member is given a nonzero value, it is used for the effective length in the perpendicular plane. Otherwise, the analysis of a laterally unsupported member will use the analog length of a Member or 0.8 times the analog length of an Interior Member.

PPSA II can perform analyses of a wide variety of plane structures providing useful values of member end actions and displacements but it cannot detect, within itself, cases in which the automated treatments of effective member length are inappropriate. This is particularly the case when effective member lengths should be longer than the analog member length. In such cases the user must supply suitable estimates of effective length through the use of COLH and COLT if table IX values are to be utilized.

## Lateral Support of a Member

Lateral support conditions can affect the analysis of any load-carrying member. It is thus important that the lateral support of members be given careful attention in the column 31 entry in the Member Properties lines. This entry, labeled MTYPE1, offers 10 options.

The classification "Member" pertains to a main member in the structure such as a truss chord. It will likely carry primary bending loads and will often be framed as continuing through a joint. An "Interior Member" is secondary in the structure such as a truss web framing into a chord. It will likely not carry primary bending loads and, since it is usually jointed to an edge of a member, will often have a longer analog length than its actual length.

MTYPE1 classes 0 through 3 relate to visually graded lumber or any other prismatic wood member that would require the use of column formulae as given in NDS 3.7 and beam slenderness factors as given in NDS 3.3.3. MTYPE1 classes 8 through B are similar but pertain to MSR and herein use column formulae as given in NDS G.9 and beam slenderness formulae as given in NDS O.6. MTYPE1 classes 4 through 7 are for a special class of trusses as given in NDS 3.10.5. Of all of these MTYPE1 classes, 1, 3, 9, and B are not laterally supported. These may require entry of effective length BLE in columns 24-29 of the Structural Assembly lines.

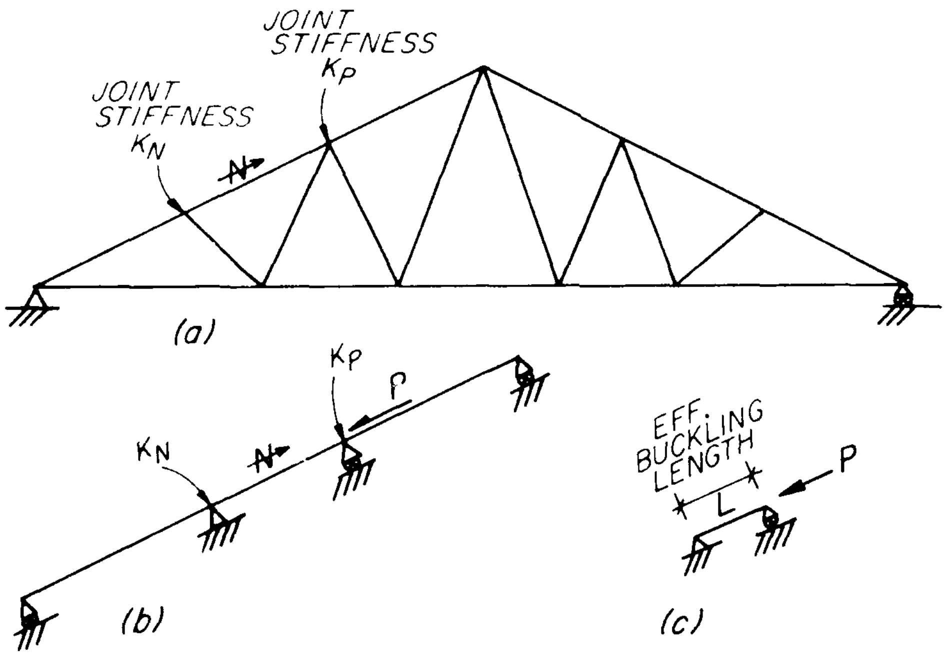



*Figure 9 - Determination of effective column length. Member N (a) is treated as the center span of an axially loaded three-span column (b). The load P is the critical buckling load for this configuration and the effective buckling length is the length of a simply supported column (c) for which P is the critical buckling load. (M151 457)*

## Interaction Indices

Values reported in table IX are governed, in part, by the ITP option selected. Interaction indices given are calculated using either equation (1), (2), or (3). These equations will handle axial tension (eq. (1)),axial compression (eq. (3)), bending (eq. (2)), tension-bending (eq. (1) or (2)), and compression-bending (eq. (3)) but are not appropriate for a member that is subject to both axial tension and axial compression. In the latter mixed case, a message MIXED TENSION-COMPRESSION, SPECIAL ANALYSIS REQUIRED is printed in lieu of the table IX line of values. The engineer can then use table VIII values for the member in constructing the shear, axial force, and moment diagrams necessary to make appropriate design decisions. It is worth mentioning that using PPSA II output values as input to microcomputer programs constructed for this purpose is a convenient way to handle such special cases. The complex arithmetic operations have already been performed within PPSA II.

For a laterally unsupported member, the interaction index given is the maximum value from calculations made both within and perpendicular to the plane of the structure. If a perpendicular plane analysis gives the largest interaction index, a reference ('***') to the footnote INTERACTION INDEX ANALYZED FOR PERPENDICULAR PLANE will appear in column 3.

If the interaction INDEX reported was calculated using equation (2) (stress difference), the footnote MAX INTERACTION INDEX IS STRESS DIFFERENCE: NDS 3.10.1 is referenced ('#') in column 3.

Values reported in columns 4-7 give the stress values used to calculate the interaction index and their location within the member selected according to the specified ITP option.

Column 8 lists the length/depth ratios calculated for each member. These ratios are compared to limiting values of 50 (NDS 3.7.2, 3.8.2) for compression members and 80 for tension members. If either of these limits is exceeded, these members are flagged by a footnote reference in column 2.

When the length/depth ratio is reported as exactly 11, the effective column length may fall into a short category, eliminating the necessity for calculating an actual value. For instance, in the uniformly loaded truss option, ITP = 3, the column length/depth ratio is set at 11 (short column) to calculate combined stress indices at panel points.

# Special Topics

The PPSA II program provides options for handling large symmetric structures or odd shapes. The following discussion describes techniques for handling these special cases.

## FIRL Reaction

The FIRL reaction behaves like the ROLL reaction with the added constraint that the reaction node is restrained against rotation. Figure 10 illustrates the comparative restraints of the two related reaction types in a simple situation. The principal use of the FIRL is to reduce the problem size when the structure and loads are symmetric except for minor reaction details.

Figure 11 provides an example of application. The beam at the top is symmetric about the vertical centerline, and all vertical displacements in the left half are identical to those in the same relative position in the right half. Also, all horizontal displacements in the structure are symmetric relative to the vertical centerline even though they are asymmetric with respect to the ground. The internal actions and displacements in half of this structure can be duplicated by using the analog containing a FIRL reaction as shown in the lower portion of figure 11. The problem size is reduced in terms of program preparation, and less computing time is required while complete information about the original structure can still be obtained. The horizontal displacements in the second analog are given with respect to node 2, which is the centerline in the original structure. Aside from this, all other necessary information is the same if the total structure is analyzed.

This example is intended to illustrate the principle and does not accurately portray the possible magnitude of savings in time and effort. Figure 12 shows a more realistic application, in which the problem size reduction is noteworthy. The symmetric truss and loading shown in the upper portion requires 13 nodes, 4 node loads, and 23 members, of which 6 carry uniform loads. By splitting the structure at the centerline and using 2 FIRL reactions at the cut, the problem now requires only 8 nodes, 2 node loads, and 12 members with 3 carrying uniform loads. The horizontal displacements must, of course, be corrected to relate the full to the half structure in this regard. The latter problem is of such minor magnitude, however, that significant advantage is realized by the use of the FIRL reactions in symmetrical structures.

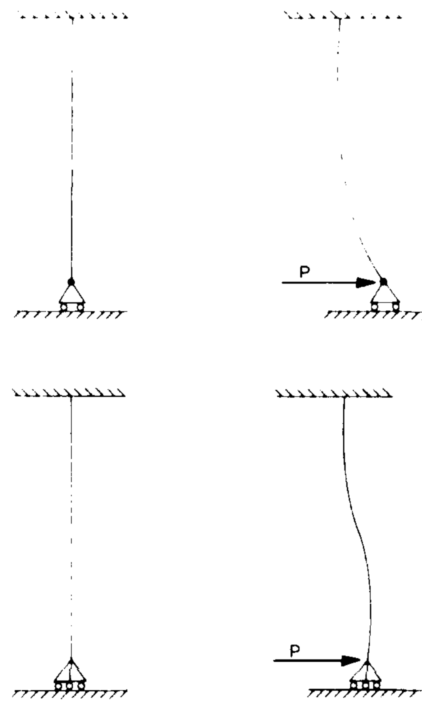



*Figure 10.--Comparison of ROLL (left) and FIRL (right) reactions. Because a member is pinned to the roll reaction and fixed to the FIRL reaction, there is no end moment at a ROLL reaction. (M139 743)*

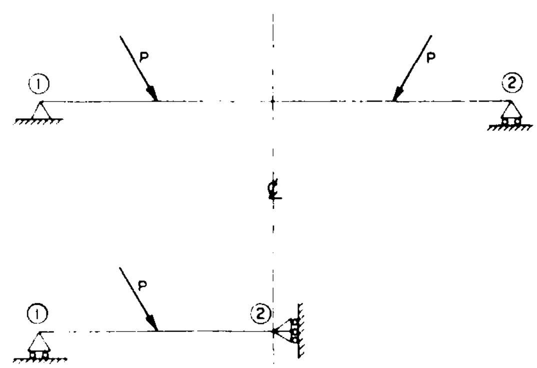



*Figure 11.--The main application of a FIRL support is in modeling symmetrical structures such as the upper simple beam. With the FIRL reaction at the center of symmetry (lower beam), the analog size can be reduced. (M139 748)*

## Further Application Flexibility

Application of this system is limited only by computer capacity and the user's creativity in terms of the structure configurations and combinations of loading that can be devised. It is only feasible at this point, therefore, to use three illustrations to stimulate a proper perspective in this regard.

An asymmetrical nonprismatic arched structure, such as might be constructed by lamination, is shown in figure 13 with a possible analog. It is established practice in structural engineering to approximate tapering members with a series of prismatic ones, a natural and easy procedure with the system described here. Once the appropriate number and size of prismatic parts have been selected, the assembly of the analog is a simple matter. The right-hand side of the structure requires more detailed consideration, however, since the real member is curved as well as tapered. Again, the matter of degree of approximation is decided by the engineer, and the actual member is simulated by a series of short, straight members of suitably varying section properties. The analog illustrated could be expanded or contracted in detail according to need. If many structures of this general type were to be designed, one of them could be analyzed several times with increasing numbers of parts in the analog to find that degree of complexity required to obtain satisfactory answers. In an analysis such as this, table IX values would be largely ignored, since the breakup of the real structure into an analog does not result in members that can be adequately treated in the automated calculation of interaction values. Member forces, moments, shears displacements, etc., from the other tables will, however, be correct for the analog and can be used in achieving the end objective of the analysis.

The second illustration is an example. Many times lumber members are scarf cut for jointing and the net section is considerably reduced adjacent to the joint connector assembly. Stresses at this critical section can be obtained from table IX by placing a short real member having reduced net section dimensions in the immediate vicinity of the analog model of the joint.

An asymmetric truss bolted to a single pole rigidly planted in the ground, as shown in figure 14, is the third illustration. An accurate analysis of this structure by traditional methods is a difficult and costly process. The computer system, on the other hand, can treat this structure with whatever degree of precision is built into the analog. One possible analog is shown in the lower portion of figure 14 with special details. The tapered pole is simulated by a series of four prismatic members of rectangular section, having section properties related to average properties at comparable portions of the pole class used. The analog length of the pole can be extended below the groundline to its fixed reaction to compensate for imperfect fixation near the surface

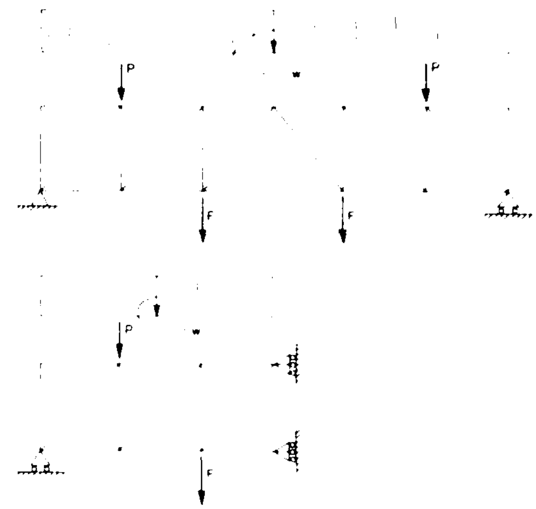



*Figure 12.--A more complicated symmetrical structure provides greater benefit to the use of the FIRL support. This figure shows a parallel chord truss and its reduced equivalent. (M139 738)*

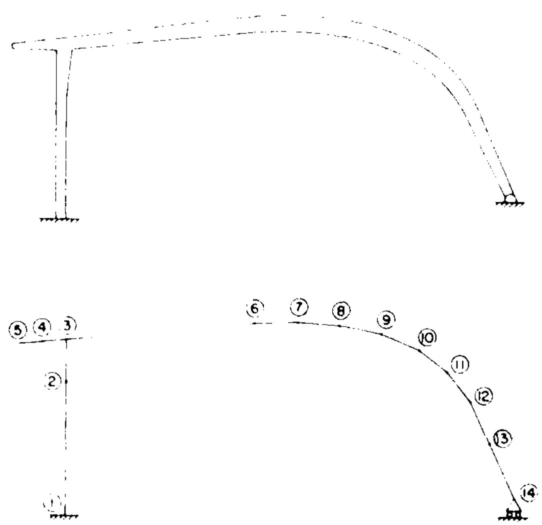



*Figure 13.--Curved and tapered frames can be approximated with prismatic segments as illustrated in the analog below the structure. The number of approximating segments can be increased or decreased according to accuracy requirements. (M139 739)*

of the earth. The analog detail in the area where the pole and the central vertical web member of the truss are superimposed is shown in figure 15. Nodes 3 and 8 are both junctions for rigid connection of truss members as shown in the upper sketch of the latter figure. The pole section (fig. 15b) is pinned to node 3, and should be pinned to node 8 but continuous in passing through this node. To accomplish this within the rules of the analytical system, node 10 is created a short distance, say 2 inches, below 8 and the pole is made up of analog members that join rigidly at 10 only. A short fictitious member is then rigidly connected to 10 and extended to 8 where it is pinned. This superimposed arrangement of members is acceptable to the computer system and reasonably simulates the mechanical behavior of the truss pole assembly.

The length of the short fictitious member in the previous example raises a question of importance to the program user. Sometimes, as in this case, shortening a fictitious member increases the duplication of reality until a limit of zero is reached. In the matrix solution used, an inversion process is performed that makes the range of member lengths from longest to shortest a potentially critical matter and the inversion cannot be completed within a reasonable time limit. In such a case, the computer keeps trying but cannot determine final answers for the structure. An increase in the length of the short fictitious member or variation of some other parameter to decrease its stiffness will correct the difficulty. Short members approximately 1/2 inch long have been successfully run with long members of over 100 inches having the same cross section and MOE ratios.

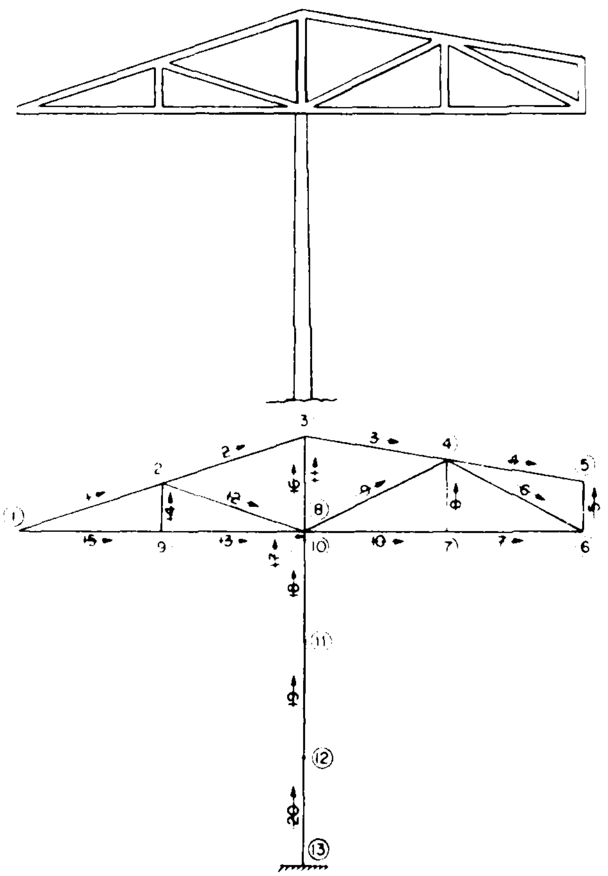


*Figure 14.--A truss with all joints sufficiently restrained to be presumed rigid is fastened to a pole with suitable timber connectors at the upper and lower chord. These connectors are presumed to act as pins. The analog drawn below shows member 11 representing the truss post, superimposed over member 16 representing the pole. Member 11 fastens to nodes 3 and 8, whereas member 16 extends from node 3 to node 10. (M139 736)*

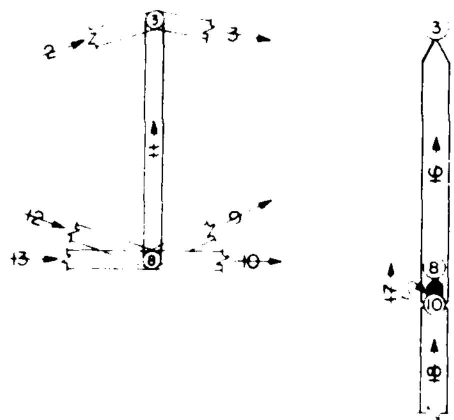


*Figure 15.--The central region of the truss analog, shown with exaggerated nodes and lines at left, has all members rigidly attached at nodes 3 and 8. Member 16, representing the pole lying behind the center post of the truss and shown at right, is pinned to node 3 but passes by node 8 and is rigidly connected to node 10 a short distance beyond. The pole immediately below the truss is represented by member 18, which is also rigidly attached to node 10. The pin-type timber connection between the pole and the lower chord of the truss is simulated by member 17 rigidly attached to node 10 and pinned to node 8. (M139 735)*

## Acknowledgments

The PPSA II program is the result of contributions by many people beginning with Gregory F. Reardon of CSIRO, Melbourne, Australia. Initial formulation and programming began during his residence at Purdue University, Lafayette, Ind., in 1966 as a visiting scientist. Since that time many others have contributed to system refinement. At Purdue Quentin B. Comus, Forrest E. Goodrick, Larry A. Beineke, Philip J. Przestrzelski, Michael H. Triche, and Frank E. Woeste are among those who have made significant contributions. Roy Adams, while a graduate student at Washington State University, performed a major reorganization of the program which greatly increased its speed of operation and made its functional understanding easier through an improved reorganization of subroutines.

This reorganized program includes condensed versions of subroutines developed at the Argonne National Laboratory, Chicago, Ill., to calculate eigenvalues and has been the foundation for further development during the past several years. The authors are also grateful for the cooperation of member companies of the Truss Plate Institute. Engineers and systems analysts in these companies have made many trial runs of developmental versions of PPSA II. Their findings and comments have led to many refinements and improvements in the program.

## Literature Cited

**National Forest Products Association.** National design specification for wood construction. Washington, DC: National Forest Products Association; 1982 edition.

**Orosz, Ivan.** Simplified method for calculating shear deflection of beams. Res. Note FPL-0210. Madison, WI: U.S. Department of Agriculture, Forest Service, Forest Products Laboratory; 1970.

**Przemieniecki, J. S.** Theory of matrix structural analysis. New York: McGraw-Hill; 1968.

**Suddarth, S. K.** A computerized wood engineering system: Purdue plane structures analyzer. Res. Pap. FPL 168. Madison, WI: U.S. Department of Agriculture, Forest Service, Forest Products Laboratory; 1972.

**Suddarth, S. K.; Percival, D. H..** Increasing the application efficiency of performance tests with analytic procedures. Performance concept in buildings. Spec. Publ. 361, vol. 1. Washington, DC: U.S. Department of Commerce, National Bureau of Standards; 1972.

**Zienkiewicz, O. C.** The finite element method in structural and continuum mechanics. New York: McGraw-Hill; 1967.

# Appendix A
# Input Data

## Identification

Any 68 characters beginning in column 1 and extending to column 68 on a single line or card.

| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 | 18 | 19 | 20 | 21 | 22 | 23 | 24 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | | | | | | | | | | | | | | | | | | | | | | | |

| 25 | 26 | 27 | 28 | 29 | 30 | 31 | 32 | 33 | 34 | 35 | 36 | 37 | 38 | 39 | 40 | 41 | 42 | 43 | 44 | 45 | 46 | 47 | 48 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | | | | | | | | | | | | | | | | | | | | | | | |

| 49 | 50 | 51 | 52 | 53 | 54 | 55 | 56 | 57 | 58 | 59 | 60 | 61 | 62 | 63 | 64 | 65 | 66 | 67 | 68 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | | | | | | | | | | | | | | | | | | | |

## Problem Size

NP = Number of analog nodes.
NM = Number of analog members.
NR = Number of ROLL reactions.
NPIN = Number of PIN reactions.
NFIR = Number of FIRL reactions.
NFX = Number of FIX reactions.
INPRIN = Control of input listing.
 0 or blank = Input is listed.
 1 = Listing of input not printed.
JPRINT = Control of output listing.
 0 or blank = Full output is listed.
 1 = Tables VII and VIII listed.
 2 = Table IX listed.
 3 = Tables IX and X listed.
 4 = Tables XI and XII listed.
 5 = Tables IX-XII listed.
NLOAD = Number of load cases for complete analysis.
ND = Number of locations checked for stress and deformation along each nonfictitious member.

| NP | | NM | | NR | | NPIN | | NFIR | | NFX | | INPRIN | JPRINT | NLOAD | | ND | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 | 18 |
| | | | | | | | | | | | | | | | | | |

## Member Properties

Member group number is right justified in field.
MTYPE1 is member type.
0 = Member, laterally supported as by sheathing, etc.
1 = Member, not laterally supported.
2 = Interior member, laterally supported as by adequate bracing.
3 = Interior member, not laterally supported.
4 = Truss chord, seasoned (NDS 3.10.5).
5 = Truss chord, green (NDS 3.10.5).
6 = MSR truss chord, seasoned (NDS 3.10.5).
7 = MSR truss chord, green (NDS 3.10.5).
8 = MSR member, laterally supported.
9 = MSR member, not laterally supported.
A = MSR interior member, laterally supported.
B = MSR interior member, not laterally supported.
NOM is last line indicator (1 for last line; blank otherwise).

Note: For fictitious members, leave allowable stresses and MTYPE blank. For fictitious members to include shear deflection, set allowable landing stress at 9000.

| Member Group No. | | Member Thickness | | | | | Member Depth | | | | | Modulus of Elasticity | | | | | | Allowable Bending Stress | | | | Allowable Compression Stress | | | | Allowable Tensile Stress | | | | MTYPE1 | NOM |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 | 18 | 19 | 20 | 21 | 22 | 23 | 24 | 25 | 26 | 27 | 28 | 29 | 30 | 31 | 32 |
| | | | | | | | | | | | | | | | | E | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | E | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | E | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | E | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | E | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | E | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | E | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | E | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | E | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | E | | | | | | | | | | | | | | | |

## Stress Factor Options

Three stress adjustment factors may be specified for use with current and subsequent loading conditions placed on the same structural analogue.

| FACTOR I | | | | | FACTOR II | | | | | FACTOR III | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 |
| | | | | | | | | | | | | | | |

## Node Coordinates

Node number is right justified.
LAST is last line indicator (1 for last line; blank otherwise).

| Node No. | | COORDINATES | | | | | | | | | | | | | | LAST |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | | x | | | | | | | y | | | | | | | |
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |

## Structural Assembly

Member number is right justified.
LAST is last line indicator (1 for last line; blank otherwise).

| Member No. | | Neg. End Node | | Pos. End Node | | Member Group | | End Fixation | | LAST | Effective Column Lengths (Engineer's Override Option) | | | | | | | | | | | | Effective Bending Length (in.) | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | | | | | | | | Neg. | Pos. | | In-Plane (in.) | | | | | | Perp. to Plane (in.) | | | | | | | | | | |
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 | 18 | 19 | 20 | 21 | 22 | 23 | 24 | 25 | 26 | 27 | 28 |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | | | | | | |

## Reaction

Reaction node number is right justified.
Reaction type is PIN, ROLL, FIX, or FIRL.
LAST is last line indicator (1 for last line; blank otherwise).

| Reaction Node No. | | Reaction Type | | | | Horizontal Component | | | | | Vertical Component | | | | | LAST |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | |

## Load Information and Interaction Interpretation

ICON = 1 if concentrated loads anywhere; 0 otherwise.
IUN = 1 if uniform loads anywhere; 0 otherwise.
ISYST = 1 if loads at node anywhere; 0 otherwise.
ITP = Interaction equation interpretations per text instructions.
SFI = Stress Factor Index--value corresponding to order of stress factor input.

| ICON | IUN | ISYST | ITP | SFI |
|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 |
| | | | | |

## Concentrated Loads

Member number is right justified.
NEXT is last line indicator (1 for last line; blank otherwise).

| Member No. | | Load No. | Horizontal Component | | | | | | Vertical Component | | | | | | Load Dist. from Neg. End (in.) | | | | | | NEXT |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 | 18 | 19 | 20 | 21 | 22 |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | |

## Uniform Loads

Member number is right justified.
NIX is last line indicator (1 for last line; blank otherwise).

| Member No. | | Horizontal Component | | | | | | | | | | Vertical Component | | | | | | | | | | NIX |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 | 13 | 14 | 15 | 16 | 17 | 18 | 19 | 20 | 21 | 22 | 23 |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |
| | | | | | | | | | | | | | | | | | | | | | | |

## System Coordinate Loads

Node number is right justified.
Load direction
  1 = horizontal.
  2 = vertical.
  3 = moment.
NON is last line indicator (1 for last line; blank otherwise).

Note 1. For multiple load cases, repeat data input from and including the Load Information and Interaction Interpretation line to this point.
Note 2. Last line of data must contain ENDDCALC starting in column 1.

| Node No. | | Load Direction | Load (lb. or in.-lb.) | | | | | | | | NON |
|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 | 11 | 12 |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |
| | | | | | | | | | | | |

# Appendix B
# Program Modification For Problem Size

There are four primary variables used to determine the storage capacity required by PPSA II to handle any given problem. They are the maximum number of members (NM), and maximum number of nodes (NP), the maximum number of reactions (NR), and the maximum number of different member groups (MG) to be assigned. Along with these primary values, there are also two less important limiting values that may require modification: the maximum number of locations along each member checked for stress and deformation (ND), and the maximum matrix bandwidth (MBW). Table B-1 lists these variables along with the maximum values allotted in the latest version of PPSA II.

The maximum value currently allowed for MBW is 30. This value limits the spread between the end node numbers of any member to (MBW + 3)/3 or 11. A problem requiring a node spread greater than 11 will rarely be encountered. Therefore, there should be little or no need to modify this value.

If any of the maximum values listed in table B-1 must be changed, tables B-2 and B-3 list the accompanying storage allotment changes required and tell where the affected variables may be found. Storage allocations are declared in common block statements (BEE1-BEE12) and dimension statements listed at the beginning of each program subunit. Modifications listed in tables B-2 and B-3 must be made in each of the subroutines indicated.

**Table B-1.--Storage requirements for arrays**

| Variable | ID[1] | Maximum value |
|---|---|---|
| Number of members | NM | 60 |
| Number of nodes | NP | 45 |
| Number of member groups | MG | 15 |
| Bandwidth | MBW | 30 |
| Number of reactions | NR | 16 |
| Number of divisions/ member + 1 | ND | 25 |

[1]Variable reference used in tables B-2 and B-3.

Table B-2.--Program array modifications required to increase the number of analog members (NM) and/or number of points (NP)

| Program parameter changed | Location: Common | Location: Other | Arrays affected | Main | Loau | Action | Inout | Member | Intact | Defl | Stiff | Symsol | Calcap | Trnslt | Colum |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| NM | BEE2 | | R(NM.2).RZ(NM.4) | X | X | X | X | X | | X | X | | | X | |
| | BEE3 | | P(NM.6) | X | X | X | | X | | X | | X | | X | |
| | BEE4 | | JAY(NM).JAYJAY(NM) | X | X | X | X | | | | | | | X | X |
| | BEE5 | | AL(NM).E(NM).EA(NM) H(NM).T(NM).C3(NM) C6(NM).EI(NM) COLT(NM). COLH(NM) BLE(NM) | X | X | X | X | X | X | X | | | X | | X |
| | BEE7 | | SIGC(NM).SIGM(NM) SIGT(NM).SIGMP(NM) PO(NM.6) | X | X | X | X | X | X | | | | | | |
| | BEE8 | | Q(NM.3.2). W(NM.2). XQ(NM.3). | X | X | | | X | | | | | | | |
| | BEE9 | | AA(NM.3.2) WMA(NM.2) PF(NM.6).ALA(NM) MEM(NM).QMA(NM.3.2) | X | | X | | X | X | X | | | | | X |
| | BEE10 | | MTYPE(NM) | | | | | | | | | | | | |
| | | Dimension | EKS(NM).TAU(NM) | | | | | | X | | | | | | |
| NP | BEE1 | | K(3NP.MBW) | X | | X | | X | X | | X | X | X | | X |
| | BEE2 | | UU(NP.3) BETA(6.3(NP-1)). | X | X | X | X | X | | X | X | | | X | |
| | BEE3 | | U(3(NP-1)). F(3(NP-1)). FF(3(NP-1)). AFF(NP.3) | X | X | X | X | X | | X | X | | | X | |
| | BEE4 | | RL(3NP).CRL(3NP) | X | X | X | X | | | | | | | X | X |
| | BEE7 | | XM(NP.2) | X | X | X | X | X | X | | | | | | |
| | BEE8 | | FO(3(NP-1)) | X | X | | | X | | | | | | | |
| | | Dimension | HR(NP).RT(NP).VR(NP) | | | X | | | | | | | | | |
| | | Dimension | CS(3(NP-1)) | | | | | | | | | X | | | |
| | | Dimension | MEMCON(NP) | | | | X | | | | | | | | |
| | | Integer | CONFLG(NP) | | | | X | | | | | | | | |

**Table B-3.--Program array modifications required with a change in maximum number of reactions (NR), maximum number of different member groups (MG), maximum number of locations checked for stress and deformation (ND), and maximum bandwidth (MBW)**

| Program parameter changed | Location | | Arrays affected | Subroutines containing affected arrays | | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | Common | Other | | Main | Load | Action | Inout | Member | Intact | Defl | Stiff | Symsol | Calcap | Trnslt | Colum |
| NR | BEE7 | | NRE(NR) | X | X | X | X | X | X | | | | | | |
| MG | | Dimension | AG(MG),AH(MG) | | | | | | | | | | | | |
| | | | AT(MG) | | | | | | | | | | | | |
| | | | ASIGC(MG) | | | | | | | | | | | | |
| | | | ASIGM(MG) | | | | X | | | | | | | | |
| | | | ASIGT(MG) | | | | | | | | | | | | |
| | | | MTYPE1(MG) | | | | | | | | | | | | |
| ND | BEE6 | | XD(ND) Y(ND) | | | | | | | X | | | | | |
| | | Dimension | X(ND) FX(ND) | | | | | | X | | | | | | |
| MBW | BEE1 | | K(3NP,MBW) | X | | X | | X | X | | X | X | X | | X |
| | | | KBW = MBW | X | | | | | | | | | | | |

2.0-5/84



☆U.S. GOVERNMENT PRINTING OFFICE: 1984 654 025 4040

DAT
FILM